

Citation for published version:

Tuechler, S & Copeland, C 2020, 'Validation of a Numerical Quasi One-Dimensional Model for Wave Rotor Turbines with Curved Channels', *Journal of Engineering for Gas Turbines and Power*, vol. 142, no. 2, 021017. <https://doi.org/10.1115/1.4044286>

DOI:

[10.1115/1.4044286](https://doi.org/10.1115/1.4044286)

Publication date:

2020

Document Version

Peer reviewed version

[Link to publication](#)

Publisher Rights

CC BY

(C) ASME 2019

University of Bath

Alternative formats

If you require this document in an alternative format, please contact:
openaccess@bath.ac.uk

General rights

Copyright and moral rights for the publications made accessible in the public portal are retained by the authors and/or other copyright owners and it is a condition of accessing publications that users recognise and abide by the legal requirements associated with these rights.

Take down policy

If you believe that this document breaches copyright please contact us providing details, and we will remove access to the work immediately and investigate your claim.

Validation of a Numerical Quasi One-Dimensional Model for Wave Rotor Turbines with Curved Channels

Stefan Tüchler* and Colin D. Copeland

Powertrain and Vehicle Research Centre

Department of Mechanical Engineering

University of Bath

United Kingdom

Email: S.Tuechler@bath.ac.uk

A wave rotor is a shock-driven pressure exchange device that, whilst relatively rarely studied or indeed, employed, offers significant potential efficiency gains in a variety of applications including refrigeration and gas turbine topping cycles. This paper introduces a quasi one-dimensional wave action model implemented in MATLAB for the computation of the unsteady flow field and performance characteristics of wave rotors of straight or cambered channel profiles. The purpose here is to introduce and validate a rapid but reliable method of modelling the performance of a power-generating wave rotor where little such insight exists in open literature. The model numerically solves the laminar one-dimensional Navier-Stokes equations using a two-step Richtmyer TVD scheme with minmod flux limiter. Additional source terms account for viscous losses, wall heat transfer, flow leakage between rotor and stator endplates as well as torque generation through momentum change.

Model validation was conducted in two steps. First of all, unsteady and steady predictive capabilities were tested on threeport pressure divider rotors from open literature. The results show that both steady port flow conditions as well as the wave action within the rotor can be predicted with good agreement. Further validation was done on an in-house developed and experimentally tested four-port, three-cycle, throughflow micro wave rotor turbine featuring symmetrically cambered passage walls aimed at delivering approximately 500 W of shaft power. The numerical results depict trends for pressure ratio, shaft power and outlet temperature reasonably well. However, the results also highlight the need to accurately measure leakage gaps when the machine is running in thermal equilibrium.

Nomenclature

Nu Nusselt number

Pr Prandtl number

Re Reynolds number
a Speed of sound in m/s
A Cross sectional area in m^2
C Absolute velocity in m/s
 c_p Specific heat constant at constant pressure in J/kg-K
 c_v Specific heat constant at constant volume in J/kg-K
 C_D Leakage Coefficient
 C_f Friction multiplier
D Diameter in m
e Internal energy in J/kg
E Total energy per unit volume in J/ m^3
f Friction factor
F Flux vector, Force per unit volume in N/ m^3 , non-dim. viscosity parameter
G Non-dim. leakage parameter in m
h Heat transfer coefficient in W/ m^2 -K, specific enthalpy in J/kg, channel height in m
k Surface roughness value in m, thermal conductivity W/m-K
m Meridional coordinate
 \dot{m} Mass flow rate in kg/s
p Pressure in Pa
 \dot{Q} Heat generation rate in W
q Heat flux in W/ m^2
r Radius in m
x Spatial coordinate
s Source term, specific entropy in J/kg-K
t Time in s
T Temperature in K, non-dim. gradual passage opening parameter
u Velocity in m/s
U State vector, tangential velocity in m/s
V Volume in m^3
W Relative velocity in m/s
 δz Cell size in m
 α Absolute flow angle in rad, under-relaxation factor
 β Relative flow angle in rad

*Address all correspondence to this author.

γ Ratio of specific heats
 δ Axial leakage gap in m
 ζ Loss coefficient
 θ Azimuthal angle in rad
 Θ Leakage function
 λ Loop flow ratio
 μ Dynamic viscosity in kg/m-s
 ρ Density in kg/m³
 τ Shear stress in N/m², time constant in s
 ω Angular velocity in m/s
 CFD Computational fluid dynamics
 EGR Exhaust gas recirculation
 FAE Fresh air exhaustion
 HPA High pressure gas
 HPG High pressure gas
 LPA Low pressure air
 LPG Low pressure gas
 TVD Time variation diminishing

INTRODUCTION

Over the past decades numerous research studies have been dedicated towards the investigation of dynamic pressure exchange machinery, such as wave rotors. These devices use the energy carried in moving shock waves to transfer energy from one stream of fluid to another without the need to incorporate additional mechanical parts. This advantage, in combination with relatively large pressure ratio gains through shock wave compression and a high efficiency associated with this process, renders wave action devices an attractive technology for power generation.

Opposed to crypto-steady flow devices, such as turbomachinery, wave rotors are inherently unsteady flow devices, where shock and expansion waves travel along discrete channels arranged around the circumference of a cylindrical drum. To each side of the spinning rotor, there are stator endplates containing port openings, as shown in Figure 1. Exposing the rotor channels periodically to the ports then triggers shock and expansion waves.

The application range for wave rotors outlined by literature is diverse. The bulk of early studies focused on pressure exchangers with straight passage profiles for gas turbine topping cycles [1–8] and supercharging devices for internal combustion engines [9–16]. In recent years the application to refrigeration cycles [17–19] and pressure-gain combustors [20–22] has come into the focus of consideration.

In comparison, little amount of attention has been given to wave turbines with cambered passage walls aimed at acting both as pressure exchangers while producing shaft power through momentum change of the flow [23–27]. Documentation of these endeavours is unfortunately fragmentary. The best documented and most successful example of a wave rotor engine was done by Pearson [23, 24] at the University of Bath. Initial tests were conducted on a single cycle, through-flow wave rotor with helical passage shape leading to a power output of around 26 kW at a rotational speed of 18,000 rpm. Further research was suspended after the engine was destroyed due to overspeeding. Further experiments by

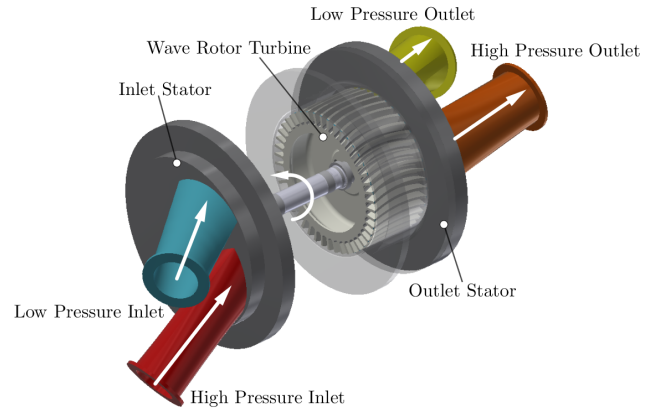


Fig. 1: Wave rotor structure showing wave rotor with discrete passages, in- and outlet stator as well as the arrangement of ports.

General Electric and General Power Corporation yielded insufficient shaft power generation and were not further pursued [25, 26].

One limitation of early efforts on wave rotor design was partially due to a lack in computational power rendering performance estimation time consuming (in particular when done by hand calculations) and often inaccurate. To mitigate this and to accurately compute unsteady wave action in the rotor channels and steady conditions in the ports as well as wave rotor performance parameters, one-dimensional codes have established themselves as swift and reliable tools. This has been pursued by a number of institutions, ranging from the Naval Postgraduate School [28–32], University of Tokyo [6, 33, 34], ONERA [6, 35, 36], Michigan State University [37], NASA Glenn Research Center [38–42] as well as Xiamen University/Beihang University [43, 44] most recently.

While most of the mentioned studies deal with straight channels, there is merely one study that incorporates passage curvature in a one-dimensional environment, albeit at reduced order through a passage-averaged description. This was conducted at NASA Glenn Research Center by Welch and Paxson [42] and compared port axial and tangential velocities as well as predicted power output to a 2D CFD simulation. However, no comparison with experimental data was done.

Against this background, this paper aims at addressing this gap by introducing a one-dimensional model that allows a reliable and cost effective insight into the performance of wave rotors with straight and arbitrarily shaped camber. To the authors' knowledge, this paper will present, for the first time, a quasi-one dimensional model that allows direct computation of torque output for cambered wave rotor channels without the need for further model reduction. It is also the first time such a model has been compared to experiments. Conventional 1D models for turbines and compressors employ either a map-based approach that requires a priori experimental data in form of mass flow rate, efficiency and pressure ratio [45] or model the rotor wheel through an adiabatic

pressure loss that is calibrated over the anticipated flow range against experimental data [46]. The proposed model, however, differs to that approach as directly models the unsteady wave action dynamics within the rotor while accounting for finite passage opening, friction, leakage and inviscid forces.

The structure of the paper is thus as follows: firstly, the governing equations and model source terms for viscous and inviscid forces, flow leakage and wall heat transfer are introduced; secondly, steady and unsteady validation of the code is done based on experimental data from the open literature on pressure dividers done by Kentfield [47] and NASA [48]. Further validation of the model is achieved through experimental data from a wave rotor turbine experiment performed in the gas stand at the University of Bath. The wave rotor turbine features a symmetrical, arc shaped passage design that was designed to produce a power output of up to 500 W.

SIMULATION MODEL

The model was implemented in MATLAB R2017 and follows a single wave rotor passage of constant cross section as it passes ports along the circumference. It consists of numerical routines to solve the one-dimensional conservation equations accounting for channel curvature for shaft power extraction, heat transfer between fluid and rotor walls, flow leakage in the axial clearance between stator and rotor, as well as gradual passage opening effects. The code can be applied to both through-flow and reverse-flow wave rotor applications.

Finally, the user has the option of defining the gas composition for the operating medium. Throughout this study this is assumed to be air, composed of 79% nitrogen and 21% oxygen.

For the derivation of the model, a number of assumptions are made; first of all, in order to justify a one-dimensional formulation, channel length is expected to be an order of magnitude larger than channel width and height. Furthermore, the flow path follows a single streamline through the passage. The quasi one-dimensional formulation assumes that all quantities are uniform across the cross section. This does not hold for wave rotors, as high rotational speeds due to centrifugal effects, vortices due to finite passage opening effects and shock-wave boundary layer interactions create secondary flows and skewed air/gas demarkation surfaces, which cannot be captured using one-dimensional wave action codes. While the effect of gradual passage opening on the primary shock strength can at least be qualitatively captured, interferences between neighbouring channels and between the channel and leakage cavity, which become particularly pronounced for larger axial clearances between rotor and stator, cannot be depicted directly.

Governing Equations

The equations used to describe the unsteady, compressible and viscous effects taking place within a wave rotor turbine are the one-dimensional Navier-Stokes equations, which can be written in conservative form as

$$\frac{\partial \mathbf{U}}{\partial t} + \frac{\partial \mathbf{F}(\mathbf{U})}{\partial m} = \mathbf{S} \quad (1)$$

The first term represents time dependent variations, while the second term refers to advection. The source term \mathbf{S} accounts for viscous as well as inviscid effects (i.e. friction and "blade" forces), as well as leakage losses and wall heat transfer. These will be explained in more detail in the next section. The state vector \mathbf{U} and the flux vector \mathbf{F} are defined as

$$\mathbf{U} = \begin{pmatrix} \rho \\ \rho u \\ \rho E \end{pmatrix}, \quad \mathbf{F} = \begin{pmatrix} \rho u \\ \rho u^2 + p - \tau_{mm} \\ \rho u \left(E + \frac{p}{\rho} - \tau_{mm} + q_m \right) \end{pmatrix} \quad (2)$$

Heat conduction within the fluid is addressed through Fourier's law of heat conduction. Furthermore friction between particles is included through shear force. They can be more explicitly stated as

$$q_m = -k \frac{\partial T}{\partial m}, \quad \tau_{mm} = (2\mu + \lambda) \frac{\partial u}{\partial m} \quad (3)$$

The set of PDEs encompasses four primitive variables u, ρ, p and e , which requires the introduction of an additional equation to ensure closure of the equation system. This is done by the ideal gas equation relating pressure, temperature and density.

$$\frac{p}{\rho} = RT \quad (4)$$

The model further treats air as a calorically imperfect gas, where specific heat constant at constant pressure $c_p = f(T)$ varies with temperature. While this is less important for pressure exchangers, where temperatures can be close to ambient conditions, it becomes more important at elevated temperatures exhibited in gas turbines, where the peak cycle temperature delivered from the combustor exceeds 450-500 K [49]. Thermodynamic data for specific heat constant and internal energy used throughout this study stems from GRI-MECH [50] and the thermal database provided by Virginia Tech [51]. Finally, dynamic viscosity is modelled through the well known Sutherland relation.

Source Terms

Viscous and Inviscid Forces

Previous one-dimensional models addressed merely viscous (friction) forces and did not directly address inviscid profile forces that account for shaft power generation. Fluid

viscosity determines wall friction and promotes convective heat transfer. As a result, the corresponding source term affects both the momentum and energy equation. Friction force is defined as

$$F_{f,res} = -C_f \rho \frac{2f}{D_h} |u|u \quad (5)$$

It features a friction multiplier C_f that account for additional momentum losses that cannot be captured with the relatively simple approach taken. As stated by Winterbone and Pearson [52] the friction factor f is a function of the Reynolds number within the channel and is given by

$$f(Re) = \begin{cases} \frac{0.25}{\left[\log_{10} \left(\frac{k}{3.7D_h} + \frac{5.74}{Re^{0.9}} \right) \right]^2} & \text{for } 5000 \geq Re \geq 10^8 \\ \frac{64}{Re_D} & \text{for } Re < 5000 \end{cases} \quad (6)$$

where k denotes the surfaces roughness value, which was set to $25\mu\text{m}$. Using the force diagram shown in Figure 2(c) one can work out the tangential component of the friction factor and write the total source term as

$$\mathbf{S}_f = \begin{pmatrix} 0 \\ s_{f,2} \\ s_{f,3} \end{pmatrix} = \begin{pmatrix} 0 \\ -C_f \rho \frac{2f}{D_h} |u|u \\ \rho \frac{2f}{D_h} |u|c_p(T_w - T) - C_f \rho \frac{2f}{D_h} |u|u \frac{u_\theta}{u} \omega r \end{pmatrix} \quad (7)$$

Torque generation in wave rotors is pulsatile in nature and is primarily generated when the channels are exposed to a high pressure inlet port. Throughout this study tangential forces and thus power generation is computed as a result of the momentum change as the fluid travels along the cambered channel walls. Overall, this is exemplified by the change in velocity triangles at leading and trailing edge of a passage wall, as shown in Figure 2(a). The corresponding blade force source term affects momentum and energy equation and takes the form

$$\mathbf{S}_v = \begin{pmatrix} 0 \\ s_{b,2} \\ s_{b,3} \end{pmatrix} = \begin{pmatrix} 0 \\ F_{b,m} \\ F_{b,\theta} \omega r \end{pmatrix} \quad (8)$$

The tangential blade force per unit volume can be determined using the change in momentum, which is influenced by the difference in absolute tangential velocity C_θ across leading and trailing edge, the mass flow rate through the channel \dot{m} and the channel volume V . In the discretised domain this force is calculated for each element taking the difference in the tangential velocity vector from one cell to the next into

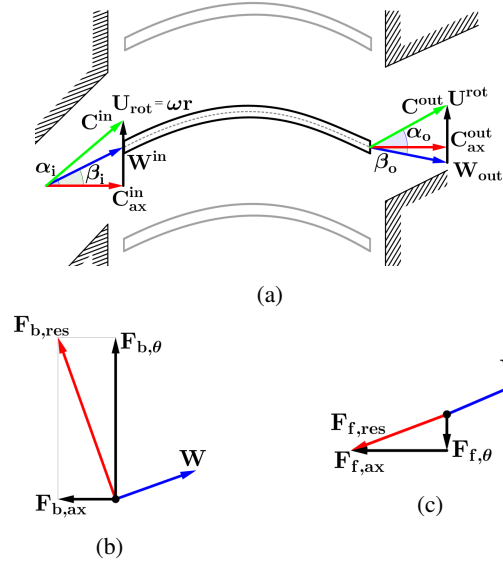


Fig. 2: (a) Velocity triangles at in- and outlet port over a curved rotor wall showing absolute velocity C , relative velocity W , tangential velocity U and axial velocity C_{ax} with the respective absolute and relative flow angles α and β . (b) Inviscid blade forces and the relative flow vector. (c) Friction vector and the relative flow angle.

account.

$$F_{b,\theta} = \frac{\dot{m}}{V} (C_{\theta,o} - C_{\theta,i}) \quad (9)$$

Friction forces are accounted for by the previously suggested viscous source term. Thus, the blade force source term acts in an inviscid way so that one can assume the blade force vector to be orthogonal to the relative velocity vector, as given in Figure 2(b). Hence, the scalar product of blade force vector and relative velocity vector equals zero and the equation can be rearranged to give the axial inviscid blade force per unit volume.

$$\vec{F}_b \cdot \vec{U} = 0$$

$$\text{so that } F_{b,ax} = -F_{b,\theta} \frac{U_\theta}{C_{ax}} \quad (10)$$

In reality, the port angles will be aligned at a different angle than the passage walls angle. This can take place at off-design conditions or if an additional momentum change is desired at the stator rotor interface in order to create more torque. In addition, flow separation on the passage wall "suction side" can often be witnessed if the port angle is considerably larger than the passage wall angle. To account for such incidence losses, an additional entropy based loss coefficient is introduced and applied in form of a distributed loss factor. It can be calibrated through experimental data or 3D-CFD simulations.

$$\begin{aligned}\Delta s &= -R \ln(1 - \zeta_{loss}) \\ F_{loss,\theta} &= \frac{\rho T |C_{ax}| \Delta s}{|U| \Delta z}\end{aligned}\quad (11)$$

And the corresponding source term is thus

$$\mathbf{S}_{loss} = \begin{pmatrix} 0 \\ s_{loss,b,2} \\ s_{loss,b,3} \end{pmatrix} = \begin{pmatrix} 0 \\ F_{loss,m} \\ F_{loss,\theta} \omega r \end{pmatrix} \quad (12)$$

The total tangential force and the corresponding shaft power generation is then computed as

$$\begin{aligned}F_{\theta,tot} &= F_{b,\theta} - F_{f,\theta} - F_{loss,\theta} \\ P_{tot} &= F_{\theta,tot} r \omega\end{aligned}\quad (13)$$

Flow Leakage

The effect of leakage is a crucial factor in wave rotor performances and was modelled as a simple, non-labyrinth leak in similar fashion as in previous publications from Kentfield, NASA and ONERA [35, 39, 53]. Leakage is modelled as a lumped capacitance model assuming steady flow equations. It concerns both continuity as well as energy equation, so that the leakage source term may be stated as

$$\mathbf{S}_v = \begin{pmatrix} s_{l,1} \\ 0 \\ s_{l,3} \end{pmatrix} = \begin{pmatrix} -C_D \sqrt{\frac{2\gamma}{\gamma-1}} \left(\frac{\delta_l}{h \Delta z} \right) \sqrt{p \rho} \Phi \\ 0 \\ -C_D \sqrt{\frac{2\gamma}{\gamma-1}} \left(\frac{\delta_l}{h \Delta z} \right) h \sqrt{p \rho} \Phi \end{pmatrix} \quad (14)$$

where C_D is the discharge coefficient, p and ρ the pressure and density in channel or cavity, h denotes enthalpy, δ_l the axial clearance between rotor and stator and Δz the cell size used for discretisation. The leakage function Φ takes up the form, which allows to differentiate between flow entering or leaving the cavity depending on the pressure ratio in cell and leakage cavity.

$$\begin{aligned}\Phi &= \sqrt{\left(\frac{p_{cav}}{p} \right)^{2/\gamma} - \left(\frac{p_{cav}}{p} \right)^{(\gamma+1)/\gamma}} \\ \text{for } \frac{p_{cav}}{p} &> \left(\frac{2}{\gamma+1} \right)^{\gamma/(\gamma-1)} \\ \Phi &= \sqrt{\left(\frac{2}{\gamma+1} \right)^{2/(\gamma-1)} - \left(\frac{2}{\gamma+1} \right)^{(\gamma+1)/(\gamma-1)}} \\ \text{for } \frac{p_{cav}}{p} &< \left(\frac{2}{\gamma+1} \right)^{\gamma/(\gamma-1)}\end{aligned}\quad (15)$$

The source terms are only active at the extremities of the rotor channel and are set to zero in the remainder of the domain. After each cycle the mass and energy balance are computed and cavity pressure and temperature updated from a first order system of differential equations as provided by Equation 16. The equations designate continuity and the energy equation derived from the first law of thermodynamics and are integrated in time using an explicit Runge-Kutta scheme. Cavity pressure can be found by applying the equation of state for ideal gases.

$$\begin{aligned}\frac{dm_{cav}}{dt} &= \sum_i \dot{m}_i \\ \frac{d}{dt}(m_{cav} e_{cav}) &= \dot{Q}_{wht} + \sum_i h_i \dot{m}_i \\ \text{where } e_{cav} &= c_v T_{cav}\end{aligned}\quad (16)$$

The heat transfer term stands for the heat transfer from the wave rotor wall temperature. For the respective heat transfer coefficient, a Nusselt number relation for annuli with inner cylinder rotation was used [54].

$$Nu = 0.015 \left(1 + 2.3 \frac{D_h}{L} \right) \left(\frac{D_o}{D_i} \right)^{0.45} Re_{eff}^{0.8} Pr^{1/3} = \frac{h D_h}{k} \quad (17)$$

Wall Heat Transfer

The viscous source term given in Equation 14 features the wall temperature to compute convective heat transfer. During operation and constant inlet temperatures from the ports the rotor temperature settles at a constant value. In order to include the effect of wall heat transfer a lumped capacitance model that follows the first-order differential equation given in Equation 18 was implemented.

$$\begin{aligned}\frac{dT_w(t)}{dt} &= -\tau^{-1} (T_w(t) - T) \\ \text{with } \tau &= \frac{m c_p}{h A}\end{aligned}\quad (18)$$

where h is the convective heat transfer coefficient, A the surface area of heat transfer and c_p the specific heat capacity of the solid material. The heat transfer coefficient h is determined from the Nusselt number relation for turbulent flow within a tube.

$$Nu = 0.0243 Re^{0.8} Pr^{0.4} = \frac{h D_h}{k} \quad (19)$$

Equation 18 is iteratively solved using a two step Heun approach, as given in Equation 20.

$$\begin{aligned}\hat{T}_{w,i}^{n+1} &= T_{w,i}^n - \alpha \left(T_{w,i}^n - T_{g,i}^n \right) \Delta t \\ T_{w,i}^{n+1} &= T_{w,i}^n - \frac{\alpha}{2\Delta t} \left(T_{w,i}^n + \hat{T}_{w,i}^{n+1} - 2T_{g,i}^n \right)\end{aligned}\quad (20)$$

Domain Discretisation

The explicit, second order accurate scheme of Richtmyer in combination with a minmod flux limiter of Roe and Baines was selected for the discretisation of the governing equations in space and time. This allows for an accurate resolution of flow discontinuities while obeying TVD criterion and preventing spurious oscillations from taking place in their vicinity. The proposed scheme suggests a two-step technique where additional half timesteps are introduced. The first step consists of a first-order accurate Lax-Friedrichs method, which can be obtained through Integration of Equation 1 in space and time and assuming the intercell fluxes to be the average of two consecutive cells.

$$\begin{aligned}\mathbf{U}_{i+1/2}^{n+1/2} &= \frac{1}{2} \left(\mathbf{U}_{i+1}^n + \mathbf{U}_i^n \right) - \frac{\Delta t}{2\Delta z} \left(\mathbf{F}_{i+1}^n - \mathbf{F}_i^n \right) - \frac{\Delta t}{4} \left(\mathbf{S}_{i+1}^n - \mathbf{S}_i^n \right), \\ \mathbf{U}_{i-1/2}^{n+1/2} &= \frac{1}{2} \left(\mathbf{U}_i^n + \mathbf{U}_{i-1}^n \right) - \frac{\Delta t}{2\Delta z} \left(\mathbf{F}_i^n - \mathbf{F}_{i-1}^n \right) - \frac{\Delta t}{4} \left(\mathbf{S}_i^n - \mathbf{S}_{i-1}^n \right)\end{aligned}\quad (21)$$

The second step uses a mid-point Leapfrog computation and consists of space and time centred differences of the mid step solutions of $\mathbf{U}_{i\pm 1/2}^{n+1/2}$ and gives

$$\mathbf{U}_i^{n+1/2} = \mathbf{U}_i^n - \frac{\Delta t}{\Delta z} \left(\mathbf{F}_{i+1/2}^{n+1/2} - \mathbf{F}_{i-1/2}^{n+1/2} \right) - \frac{\Delta t}{2} \left(\mathbf{S}_{i+1/2}^{n+1/2} - \mathbf{S}_{i-1/2}^{n+1/2} \right)\quad (22)$$

BOUNDARY CONDITIONS

For the implementation of boundary conditions a cell-centred approach has been chosen. This involves the addition of an image cell lying just outside of the domain. In general, one can distinguish between inflow, outflow and wall boundaries. For each of these types one needs to determine the flow variables on the image cells. Walls are treated as reflective boundaries where pressure and density are equal to the neighbouring interior cell node, while velocity is assigned the same value with the opposite sign.

Assuming subsonic inflow, it is necessary to specify two characteristics entering the domain, while one characteristic leaves the domain. Therefore stagnation properties (temperature and pressure) are imposed, while velocity is extrapolated in zero-th order. This approach further guarantees a simple way to compare results with experimental conditions. When computing the flow variables at the image cell, it is crucial to consider that the stagnation properties are defined in the absolute frame of reference, while the code operates in the relative frame of reference.

For outflows one characteristic enters the domain, while two characteristics leave the domain. Thus, a single physical condition needs to be imposed (static outlet pressure) and the remaining two are extrapolated from the domain interior.

Unfortunately, this is only valid for fully exposed channels. There are, however, periods where only a certain portion of the channel is exposed to a port, as shown in Figure 3(a). This gradual passage opening is of paramount importance in the formation of primary shock waves [55]. To account for this, a function is defined that determines the amount of overlap based on port opening and closing positions, channel width and the channel position at each time step.

If this function is between zero and one, the port is only partially open and an alternative solution for the boundary conditions is calculated. In case of inflow, the principle is given in Figure 3(b). The gas enters the domain from a reservoir described by stagnation properties for pressure and temperature in form of a jet through the cross section area at station 2. At station 1 the jet further expands to cover the entire width. The first cell in the domain is at station i . Assuming steady flow conditions, one can formulate the three conservative equations for mass, momentum and energy between station 1 and 2 as

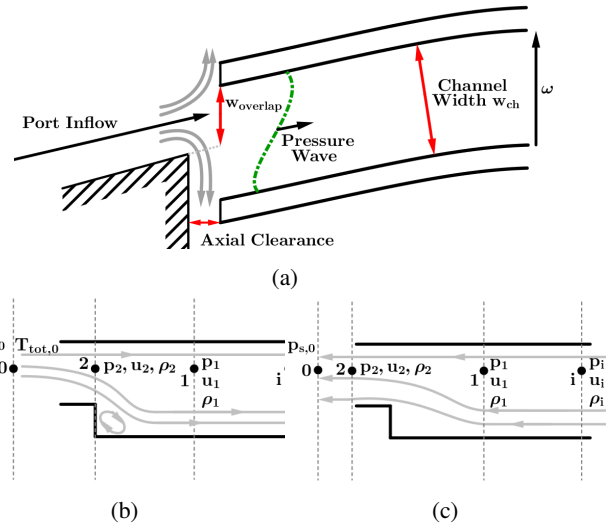


Fig. 3: (a) Gradual passage opening shown for port inflow. (b) Schematic for the boundary condition for partially open inflow (c) Schematic for the boundary condition for partially open outflow.

$$\begin{aligned}\rho_1 u_1 A_1 &= \rho_2 u_2 A_2 \\ (p_1 + \rho_1 u_1^2) A_1 &= p_2 A_1 + \rho_2 u_2^2 A_2 \\ a_{tot}^2 &= a_1^2 + 0.5(\gamma - 1)u_1^2 = a_2^2 + 0.5(\gamma - 1)u_2^2\end{aligned}\quad (23)$$

giving three equations for six unknowns in total. The set of equations is closed using the energy equation and isentropic

relation between station 0 and 1 as well as wave and pathline compatibility relations [52]. In a similar fashion, one can formulate the problem for partially open outflows, as given in Figure 3(c). Here, gas flowing through a cross sectional area of A_1 exits the domain through area A_2 . In the subsonic case the pressure at station 2 equals the reservoir pressure at plane 2. Together with the conservation of mass and energy as well as the isentropic relation between station 1 and 2 and the corresponding compatibility equations, all six unknowns can be determined.

PROCEDURE

The solution procedure of the code shall be introduced using the example of a four port throughflow wave rotor, as shown in the unfolded view of a single wave rotor cycle in the $z - t/\theta$ plane in Figure 4. The ports are designated as *HPG*, *HPA*, *LPA* and *LPG*, referring to high pressure gas and air and low pressure air and gas respectively.

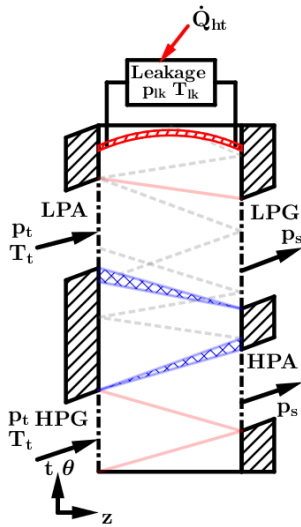


Fig. 4: Model schematic showing port arrangement and expected wave pattern for the Bath μ -wave rotor turbine. To comply with the experimental arrangement, the model was run in open loop without the inclusion of a combustor model.

It follows the steps outlined in Figure 5(b). Initially, geometric dimensions of the rotor and target rotational speed of the wave rotor are specified. Furthermore, the number of cycles per rotation and the port solution needs to be provided. An initial guess and rough layout can be devised through the analytical tools given by [43, 56–58]. Subsequently, all matrices used throughout the computation are, along with the spatial domain, initialised. In terms of the time step, an initial estimation is provided. While a uniform grid is used, the time step size is allowed to vary depending on the maximum wave speed $\max_i(|u_i^n| + a_i^n)$ in the domain at in the previous time step and a fixed CFL number of 0.7 [52]. This allows us to save computational time as otherwise a constant time

step would need to suit the largest velocities in the domain and would thus have to be more on the conservative side.

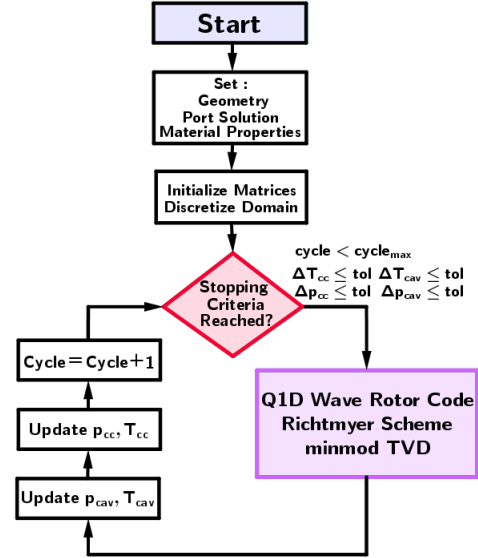


Fig. 5: Simulation model flowchart.

In open loop configuration (without a closed combustor loop), stagnation properties in the HPG port are fixed, while static pressure in the HPA port is varied automatically using a simple proportional controller based on a simple Bernoulli relation until the mass flow rates are matched and within a 2% relative error. The low pressure inlet conditions generally involve ambient temperature of around 300 K, while total inlet pressure again being varied in the same manner as for the HPA port and according to the desired loop flow ratio $\lambda = \dot{m}_{HPG} / \dot{m}_{LPA}$.

$$p^{t+1} = p^t + \alpha \frac{\rho (\dot{m}_{actual}^2 - \dot{m}_{target}^2)}{2(\rho A)^2} \quad (24)$$

In the low pressure gas port exhaust gases are expelled to the ambient and the static pressure is again automatically adjusted to give the desired inlet mass flow rate \dot{m}_{HPG} . At the end of each cycle combustor and leakage cavity properties are updated and port flow conditions need to be determined. This is done through mass flux averaging according to Equation 25 and converting the values into the absolute frame of reference.

$$\bar{\phi} = \frac{\sum_{i=N_o}^{N_c} (\phi_i \rho_i u_i)}{\sum_{i=N_o}^{N_c} (\rho_i u_i)} \quad (25)$$

Before the next cycle commences, conditions within the channel at the cycle end are set as initial conditions for the subsequent cycle, thus guaranteeing periodicity. Finally, the simulation is stopped either when the maximum number of cycles is reached or the relative changes of combustor and leakage temperature and pressures respectively are below a predefined threshold.

Results and Discussion

The results presented in the following section seek to exhibit the code's ability to predict results from wave rotor experiments. This shall be done in two stages. First of all, performance data of two three-port pressure dividers is produced and compared with data from the open literature. These are, Kentfield's pressure divider from the 1960's [47] and NASA pressure divider from the mid 1990s [48]. After that, laboratory experiments conducted at the University of Bath on a μ -wave rotor turbine are used to test the code when dealing with four port, throughflow wave rotor turbines with symmetrically cambered profiles.

Wave Rotor Pressure Dividers

The geometric dimensions and operating conditions of the two investigated designs are given in Table 1. The main characteristics of both designs shall be briefly introduced. Kentfield's device is characterised by relatively large channel width, while NASA's 3-port design has four times the number of channels and has, as a consequence, a rather small channel width. Thus, it can be expected that finite opening timing effects are of minor importance. In terms of length, the NASA design is longer, so that viscous losses become more pronounced. Leakage plays a reduced role in Kentfield's experiments owing to tighter clearances between rotor and stator endwalls.

Rotational speed of both designs and total inlet temperature in the medium pressure port are fixed in both simulations. Total inlet pressure in the medium pressure port is variable for the Kentfield experiments. In the model, the total outlet pressure of the high pressure port is varied in both cases to give a desired mass flow ratio \dot{m}_M/\dot{m}_H range from 0.1 to 0.6. The NASA experiment was conducted at a constant mass flow ratio of 0.37. The unfolded mid-plane view for the two pressure exchangers with the anticipated wave pattern and relative positions of the three respective ports is given in Figure 6. Solid red curves denote shock waves, hatched blue areas expansion fans and dashed grey lines (weak) pressure waves.

Table 1: Dimensions and operating conditions of Kentfield and NASA pressure divider used for the simulation.

Parameter	Kentfield	NASA 3-Port
Rotor diameter [mm]	164.8	294.6
Number of channels	30	120
Channel shape	straight	
Channel length [mm]	279.4	457.2
Channel width [mm]	15.2	6.4
Channel height [mm]	55.9	10.16
Nominal clearance [mm]	0.18	0.51
Number of cycles per revolution	3	1
Rotational speed [rpm]	6000	4150
T_{tM} [K]	308.3 K	353.9
p_{tM} [kPa]	110 - 150	207
p_{sL} [kPa]	100	51

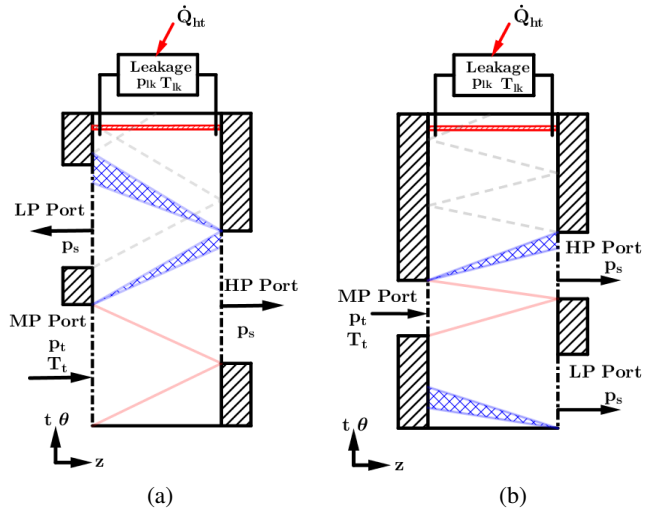


Fig. 6: Model schematic showing port arrangement and expected wave pattern for pressure dividers of (a) Kentfield and (b) NASA's Glenn Research Center.

Using the MATLAB code on Kentfield's three port pressure divider yields the results depicted in Figure 7. All data was produced with a grid resolution of $\Delta z/L = 0.0135$ and a discharge coefficient of 0.67 and a friction multiplier of 1.77. The plots show total pressure for the high pressure port on the ordinate and total pressure for the low pressure port on the abscissa for four different mass flow ratios. All values are non-dimensionalised with respect to the inlet total pressure and inlet mass flow rate respectively. Figure 7(a) shows a comparison of the 1D-Navier-Stokes equation (that include particle friction and heat conduction in the fluid but without wall heat transfer, friction and leakage model) compared with the experimental data. Clearly, the pressure level in the high pressure port is overpredicted with an increasing deviation towards lower pressure levels in the low pressure

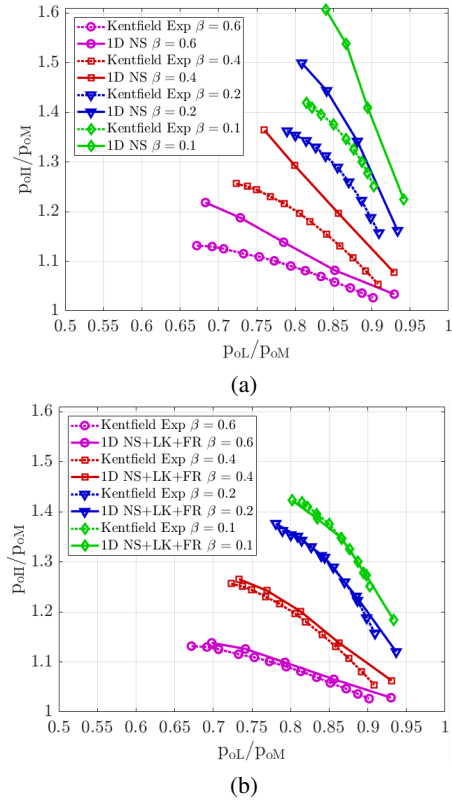


Fig. 7: Comparison of Kentfield's three-port wave rotor with the 1D simulation data. Plot shows normalised total pressure at the high pressure port versus the total pressure at the low pressure port for different mass flow ratios. (a) Illustrates simulation data for Euler equations without source terms, while (b) shows the effect of accounting for wall heat transfer, wall friction and leakage.

port. The full model results are then shown in Figure 7(b) outlining the effect losses impart. The main loss mechanisms responsible for the difference are leakage and friction, confirming the findings of previous studies [35, 39]. Since maximum temperatures in both cases are relatively low wall heat transfer plays a minor role.

The same model parameters were then applied to perform unsteady validation using the NASA three-port wave rotor. This experiment featured static pressure traces at three locations, namely at $z/L=0.025$, $z/L=0.5$ and $z/L=0.975$, within a wave rotor passage as it travels through the circumference. Evaluating the numerical results and normalised static pressure at the channel ends, namely at data against the numerical results gives the distribution shown in Figure 8. In comparison with Figure 6(b) one can witness the initial expansion fan generated upon opening the low pressure port on the right hand side. This is well captured in the model, although the expansion ratio at $z/L=0.025$ remains slightly overpredicted. The subsequent shock discontinuity is sharper than in the experiments and features a sharp peak for both primary and secondary shock waves. The overall shock pressure ratio across all stations is, however, well depicted. The final expansion fan, as well as the attenuation

of the pressure wave takes place slightly sooner, as seen at around $\theta = 3.25 - 3.5\text{rad}$ in $z/L=0.025$ and $z/L=0.5$.

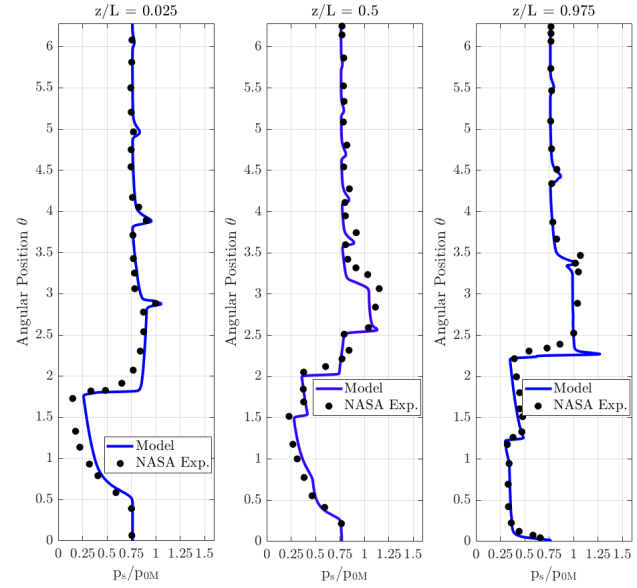


Fig. 8: Comparison of one-dimensional simulation data with experiments of a NASA three port pressure divider at $z/L=0.025$, $z/L=0.5$ and $z/L=0.975$.

Bath μ -Wave Rotor Turbine Layout and Operating Conditions

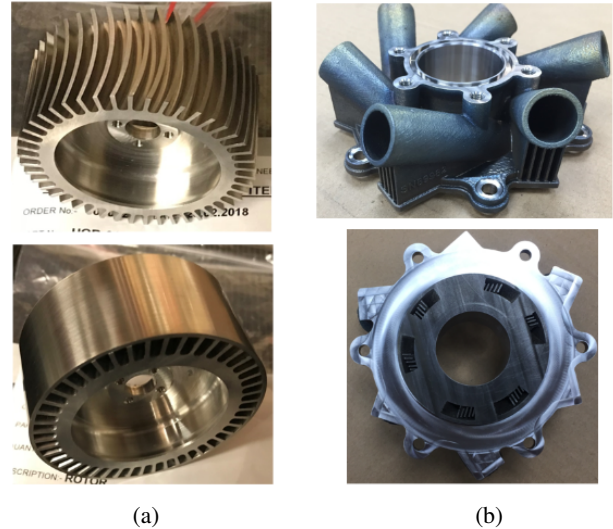


Fig. 9: (a) Wave rotor turbine with and without outer sleeve showing channel curvature. (b) Machined inlet stator exhibiting discrete port openings.

The third validation case deals with a four-port, three-cycle throughflow wave rotor with symmetrically cambered

wall profiles. The rotor was designed to yield approximately 500W of shaft power output at a target peak cycle temperature of 750°C and HPG total inlet pressure of 285 kPa. The port solution is shown in Figure 4 and can be divided into a high pressure and low pressure section in the bottom and top respectively. The former encompasses a high pressure gas inlet (HPG) and a high pressure air outlet (HPA) that would form the combustor loop in a gas turbine arrangement, while the latter houses the low pressure air inlet (LPA) and a low pressure exhaust outlet (LPG). A detailed view of the symmetrically cambered wall profiles and a photograph of the rotor and endplates with the corresponding port openings are given in Figure 9(a) and (b).

Table 3: Dimensions and operating conditions of the Bath μ -wave rotor.

Parameter	Bath μ -wave rotor
Rotor diameter [mm]	60
Number of channels	46
Channel shape	sym. cambered max. $ \beta_{ch} = 21.5^\circ$
Channel length [mm]	30
Channel width [mm]	2.9
Channel height [mm]	6
Nominal clearance [mm]	variable 0.1 - 0.4
Number of cycles per revolution	3
Design rotational speed [rpm]	32,000
T_{HPG} [K]	773-883 K
p_{HPG} [kPa]	270-290
p_{LPG} [kPa]	99

Geometric dimensions and operating conditions that will be used for the simulations are given in Table 3. The rotor has a mid-width diameter of 60 mm, length of 30 mm and features 46 symmetrically cambered channels with a maximum profile angle of 21.5° . In total, three cycles are covered in a single rotation and the design speed is set to 32,000 rpm, giving a tangential speed in the mid-width plane of 100 m/s.

The wave rotor test rig encompasses an open loop configuration, as shown in Figure 10(a). A set of industrial compressors provides pressurised and dry air to the inlet side

(HPG and LPA). The corresponding flow rates and thus the flow ratio λ are controlled through a set of pneumatically actuated gate valves and measured through differential mass flow meters. On the HPG leg, the incoming air is additionally directed through an air-to-air heat exchanger to use hot exhaust gases from the outlet side before being run through a set of 44kW-electrical heaters that ensure the target inlet temperature is reached. Rotational speed and shaft power output is measured through an eddy-current dynamometer that modulates the load on the wave rotor. It is coupled to the wave rotor through a single-plane coupling. On the outlet side, further gate valves ensure the mass flow rates in the high pressure zone, i.e. between HPG and HPA, can be matched accurately before being expelled from the test chamber through extraction fans. Figure 10(b) exhibits a photograph from the test chamber displaying the electrical heaters, as well as the wave rotor and dynamometer units and outlet gate valves.

The design of the wave rotor encompassed a variable nominal axial clearance between 0.1 to 0.4 mm. Unfortunately, it was found during the experiments that the nominal clearance differed significantly from its design value due to uneven thermal expansion. As the exact values of the actual clearances could not be determined, it was decided to use the nominal values for throughout the validation study.

Rotor Characteristics

Before moving on to the results, it is necessary to classify the wave rotor performance with respect to other existing machines. Table 2 lists a comparison of non-dimensional performance parameters for finite passage opening T , viscosity F and leakage flow G for various wave rotor types, as defined by [48] and [6]. Although comparison is somewhat difficult for pure pressure exchangers and wave rotor turbines operating at high temperatures, one can see that the Bath μ -wave rotor performs similarly with respect to finite opening timing effects despite its short rotor length. As expected, due to the small size compared with larger designs viscous losses become more pronounced and is similar to the micro wave rotor study conducted at the University of Tokyo & ONERA [6]. The most critical loss mechanism for the investigated design is without a doubt leakage effects, which is at best similar to the ABB Compress and the University of Tokyo. In reality however, it is anticipated that uneven thermal expansion of stators and the shaft-rotor assembly account for a considerably larger G -parameter approaching

Table 2: Comparison of previous wave rotors, both actually realised ones that were tested and conceptual ones (University of Tokyo/ONERA) with the Bath μ -wave rotor turbine (adapted from [6]).

	Kentfield 3-port PE	NASA 3-port PE	NASA & R.R.Allison 4-port TF	Univ. of Tokyo & ONERA micro wave rotor study	ABB Compress 4-port RF	Pearson 6-port TF wave engine	Univ. of Bath 4-port TF μ -wave turbine
T	0.39	0.082	0.19	0.31	0.467	0.3	0.37 at 32,000 rpm
F	0.0046	0.017	0.0037	0.012	0.0067	n.a.	0.013
G	0.0064	0.025-0.075	0.0090	0.067	0.030	n.a.	0.033 - 0.132

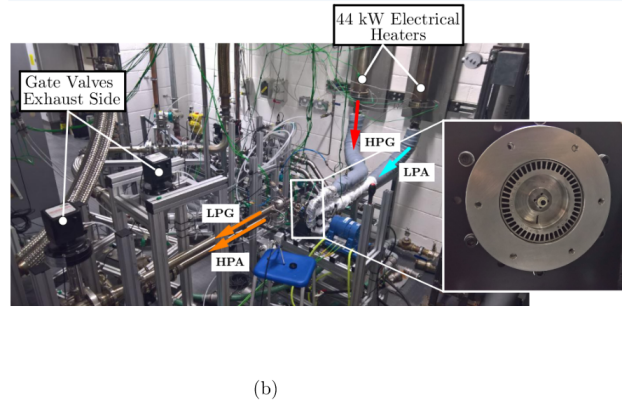
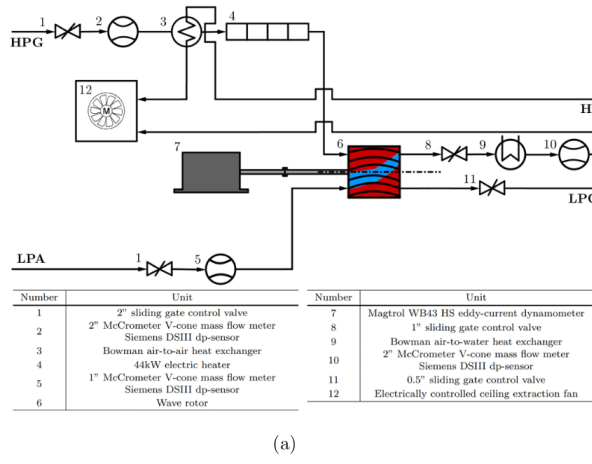


Fig. 10: (a) Schematic showing wave rotor open loop experimental setup and corresponding instrumentation. (b) Photograph taken from the test rig exhibiting the wave rotor, eddy-current dynamometer, electrical heaters and outlet side gate valves.

0.2-0.23. This effect however, cannot be captured in a one-dimensional environment and needs to be taken into account through the leakage coefficient C_D . An additional effect of leakage that cannot be identified using one-dimensional code is that increasing leakage gaps promote interaction between neighbouring channels effectively attenuating in particular the reflected secondary shock extensively.

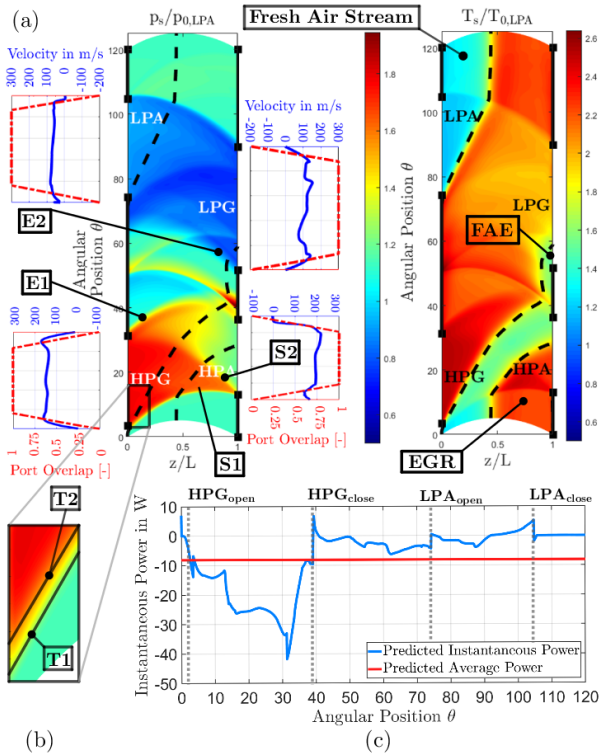
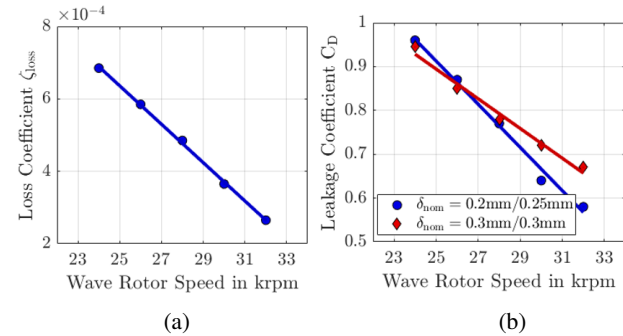


Fig. 11: Q1D numerical results showing shock wave pattern, temperature field, port velocity distribution and torque output at $\lambda = 1.7$, $N = 32,000$ rpm, $T_{t,HPG} = 500^\circ\text{C}$ and $\dot{m}_{HPG} = 0.032$ kg/s.

Fig. 12: Parameter fit for (a) loss coefficient due to channel/flow misalignment and (b) flow leakage.

Model Validation

For the first part of the experimental validation, rotational speed was swept from 24,000 rpm to the design speed of 32,000 rpm in steps of around 2,000 rpm. This is done for two different nominal leakage gaps, namely 0.20mm on the inlet and 0.25mm on the outlet side as well as 0.3mm on both in- and outlet side respectively. In addition the peak cycle temperature $T_{t,HPG}$ was varied from 500°C to 600° . The loop flow ratio λ is maintained constant at 1.7. In the second part of the validation study, λ is varied in steps of 0.5 between 1.7 to 2.7. The total inlet pressure remains at approximately 270kPa throughout.

Variations in rotational speed lead to different velocity triangles at both in- and outlet. To account for these variations and associated misalignment of the flow with the rotor profiles, a parameter fit was made for the loss coefficient ζ_{loss} . The linear relationship determined from this is given in Figure 12(a) and used throughout the validation study. The same procedure was applied to the leakage coefficient C_D . Here, the situation becomes more complicated, as different leakage gaps account for slightly different characteristics, as illustrated in Figure 12(b). The reason for this lies most likely in the fact that actual leakage gaps are not known

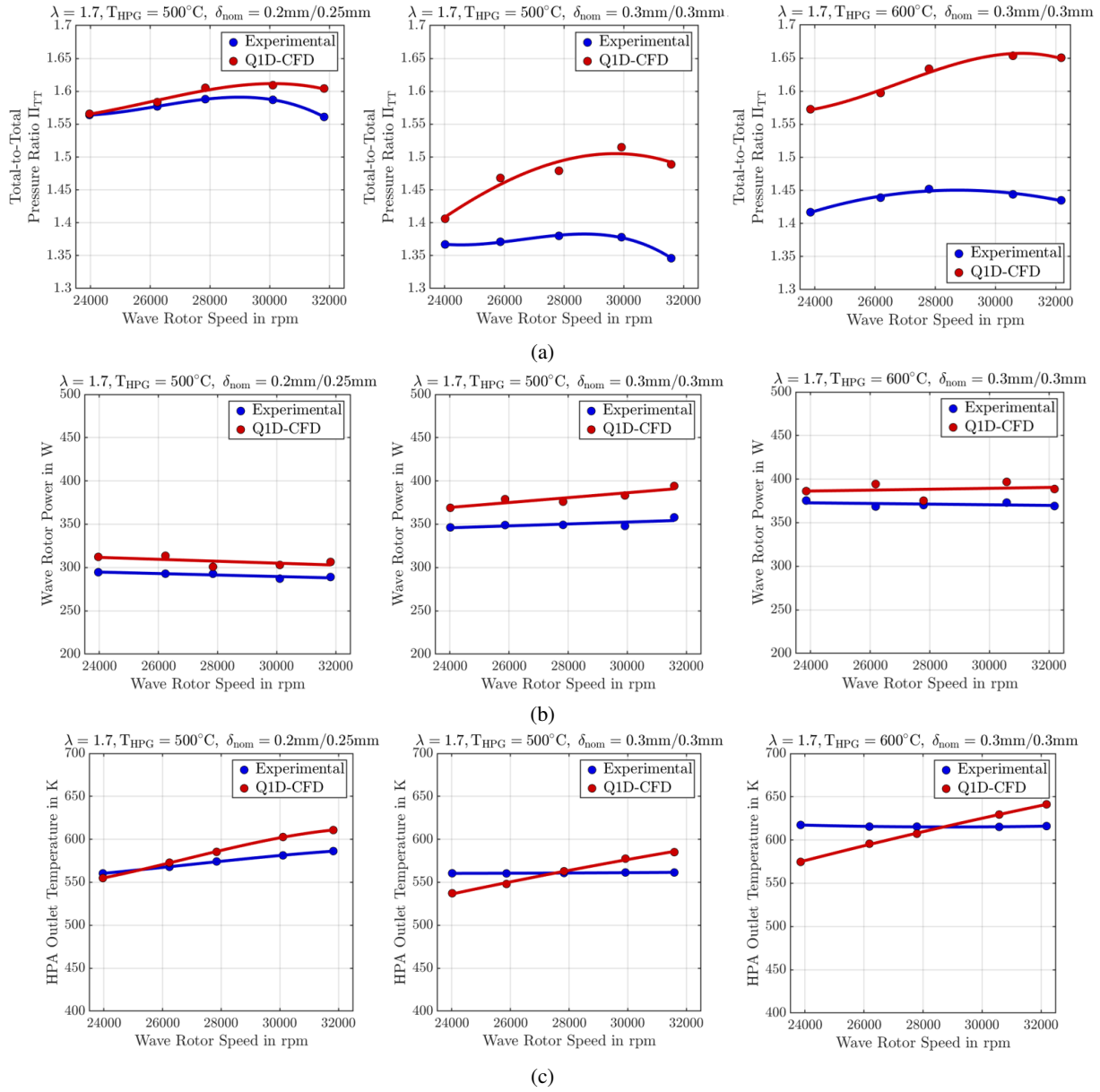


Fig. 13: Effect of leakage and peak cycle temperature variation on wave rotor performance. Comparison of experimental data and numerical model for (a) total-to-total pressure, (b) shaft power output and (c) average outlet temperature in the HPA port. The graphs on the left denote data for a loop flow ratio $\lambda=1.7$, a peak cycle temperature of 500°C and a nominal clearance gap of 0.2mm on the in- and 0.25mm on the outlet side, while graphs in the centre show the effect of increased axial gap clearance to 0.3mm and graphs on the right outline the effect of an increase in peak cycle temperature to 600°C .

and merely the nominal values were taken. Hence, the selected discharge coefficient partially accounts for this lack in information. The reason why one sees a variation of the leakage coefficient with speed can be explained by additional blockage or greater dynamic head created in the leakage gap through higher rotational speeds, to a similar as witnessed in turbomachinery [59]. Finally, the friction multiplier was set to 2.7 throughout the simulations. A similarly high value was witnessed in the studies by Paxson [39].

At the design speed of 32,000 rpm Figure 11 gives the numerically determined unfolded view of both wave pattern in the left hand side contour plot and temperature distribution

in the right hand contour plot. The plot also shows the predicted velocity profiles in each port and the extent of gradual passage opening. Marked in station S1 and S2 are primary (right running) and secondary shock (left running) waves generated upon opening of the high pressure gas (HPG) inlet and high pressure air (HPA) outlet. Further shown are expansion fans E1 and E2 produced as a result of HPG closing and LPG opening. The temperature distribution indicates inherent exhaust gas recirculation (EGR) characteristic to axial throughflow machines as well as a small amount of fresh air exhaustion (FAE) through the exhaust port LPG.

The effect of gradual passage opening is shown in Fig-

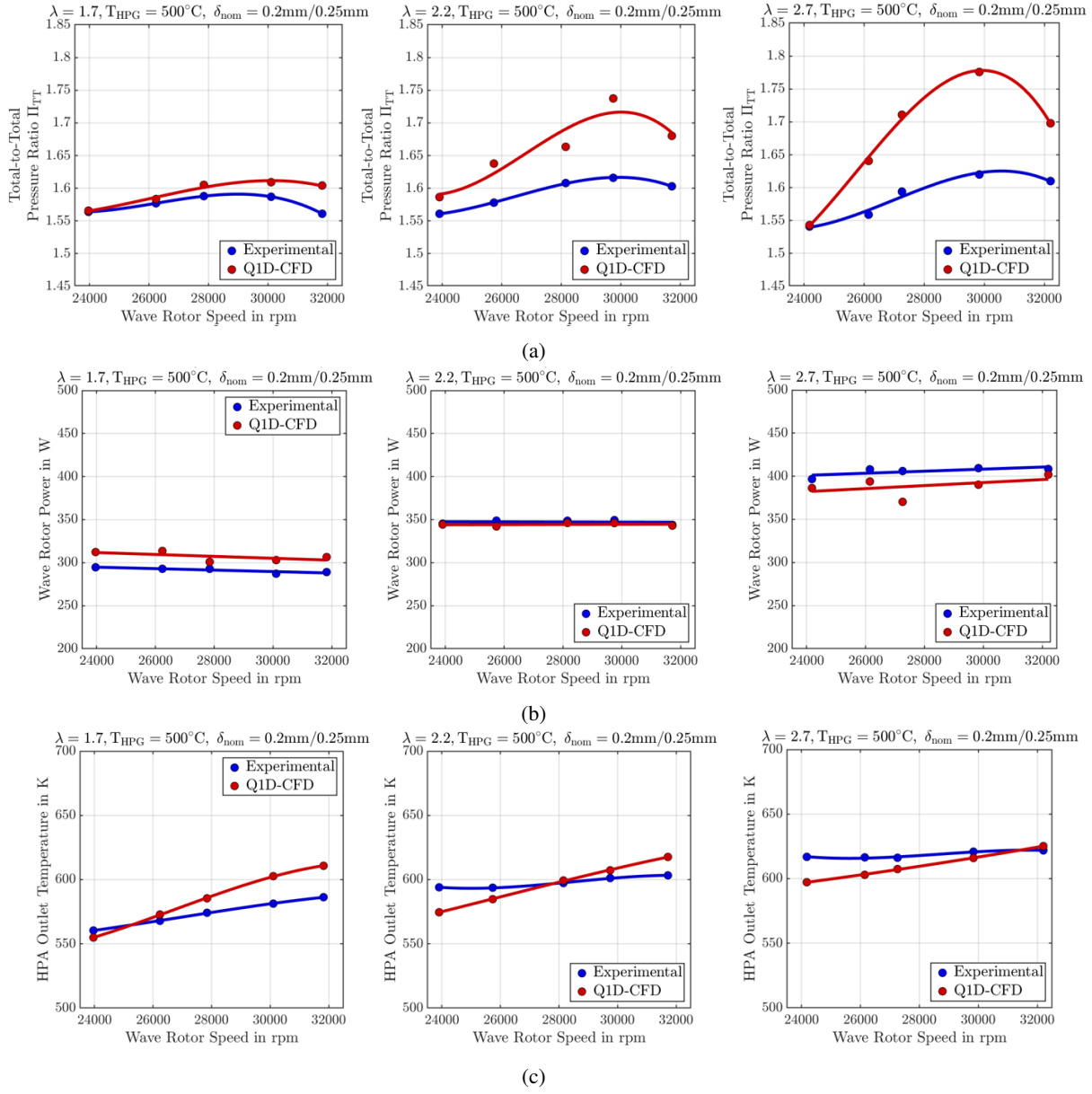


Fig. 14: Effect of loop flow ratio variation. Comparison of experimental data and numerical model for (a) total-to-total pressure, (b) shaft power output and (c) average outlet temperature in the HPA port. The graphs on the left denote data for a loop flow ratio $\lambda=1.7$, while centre plots show the results for $\lambda=2.2$ and right hand side graphs for $\lambda=2.7$.

ure 11(b), where mark T1 indicates the initiation of the primary shock wave before progressively increases towards the fully developed level at approximately T2. The predicted instantaneous and cycle-averaged power output for a single channel are given in Figure 11(c). It becomes apparent that solely the flow entering through the HPG port results in torque generation, while opening of the LPG port and inflow through LPA do not significantly contribute to overall power generation. This is of course expected as the flow through the HPG port signifies the flow with the highest enthalpy. The total expected power output is then evaluated by multiplying the number of channels with the cycle-averaged value. The plot further indicates that the proposed symmetric, arc shape is not an optimum with respect to torque generation, which is

outlined by positive regions in the instantaneous power trace near opening and closure of the HPG port.

Comparing power output, pressure ratio and outlet temperature from the HPA duct for the experimentally determined dataset with the simulation model shows the group of plots displayed in Figure 13. The first plot on the top left denotes total pressure ratio data obtained for a maximum inlet temperature of 500°C and a nominal clearance of 0.20mm on the inlet and 0.25mm on the outlet side. The model accurately predicts the total pressure distribution well across the entire speed sweep. As expected, increased leakage gaps result in compromised energy transfer and thus lower achieved pressure ratio. This trend is reflected in the model results, although there is an overprediction in pressure ratio with a rela-

tive error between experiments and simulation of around 10-15%. Furthermore, the model predicts a pronounced drop off in pressure ratio as one approaches lower rotational speeds, which has not been witnessed in the experiments. Opposed to the validation on pure pressure exchangers of Kentfield and NASA, there is a more pronounced discrepancy between simulation results and experimental data. This can at least partly be attributed to the fact that experimental work on pressure exchangers was done at nearly ambient temperature conditions. This renders thermal expansion effects minimal and axial clearances between stator and rotors are likely to remain unchanged from their nominal values. For the wave rotor turbine under investigation here, peak temperatures differ significantly from ambient conditions and uneven thermal expansion between rotor and stators causes the axial clearance gap to increase in size, which severely affects in particular the secondary shock wave strength and thus the achieved pressure ratio.

Raising maximum inlet temperature from 500°C to 600°C results in an increase in pressure ratio as a consequence of more enthalpy being directed to the wave rotor. While the overall trend is also reflected in the simulation results, the model indicates that the increase in temperature appears to outweigh the penalty in larger leakage flow, while in the experiments this is clearly not the case. This shortcoming exemplifies the sensitivity of the system that arises when choosing model factors, thus limiting its predictive capabilities.

The model yields good results with respect to predicted power output for all simulated conditions, shown in Figure 13(b). Power is slightly overpredicted throughout with a relative error of around 5-10%. The power output increase shown in (b) stems from an increase in HPG mass flow rate of 34 g/s compared to 32 g/s in the low leakage case.

Finally Figure 13(c) gives a comparison of the averaged total temperature at the HPA outlet with the experimental data recorded through thermocouples close to the port outlet. Across all tested conditions the relative error varies between 1% and 6%. While there seems to be moderate (left and right) to no (centre) variation with respect to rotational speed changes, the model predicts steeper line gradients. This is a consequence of the model predictions regarding exhaust gas recirculation at the HPA port. The temperature distribution shown in Figure 11 implies some FAE. Reducing FAE by allowing more fresh air being directed to the HPA port before the port closes. This reduces EGR rates at the HPA port and accounts for a slightly lower mixed-out temperature. This effect is well shown in Figure 13(a). The simulation results in (centre) and (right) almost show the same gradient with respect to speed as in the low leakage case in (left). The discrepancy between the two stems again from the effects of leakage, where increased leakage promotes interaction and flow between one channel, another channel and the leakage cavities around it. Thus more hot air is ingested into the cold air stream leading to more or less the same average outlet temperature.

Figure 14 exhibits the effect of a variation in loop flow ratio from 1.7, to 2.2 and 2.7 with rotational speed. The

model correctly predicts an increase in pressure ratio with increasing loop flow ratio. However, while there is a relatively small difference at a loop flow ratio of 1.7 of less than 3%, the model reacts more sensitively to an increase in loop flow ratio resulting in higher pressure ratios and increased error. Higher loop flow ratio signifies a reduced cold air mass being ingested into the wave rotor resulting in a higher average rotor temperature and thus altered thermal expansion. The Q1D-model cannot account for this change in leakage characteristic, which results in a greater error between the simulation results and the experimentally determined data.

Predicted shaft power output captures the trends of increasing power output with increasing loop flow ratio. The main mechanism for this is an increase in inlet mass flow rates to around 34 g/s at $\lambda=2.2$ and 35.5 g/s at $\lambda=2.7$. However, the increase indicated in the experiments is higher than shown for the simulation model, resulting at a maximum underprediction in shaft power at $\lambda=2.7$ of approximately 8%.

Looking at the averaged temperature in the HPA port, the maximum relative error between experiments and simulations is at around 4%. The model is able to predict the effects of the decrease in fresh air flow well. Higher loop flow ratios increase EGR and thus mark a reduced effect of fresh air temperature on the mixed out average temperature. As a result, average temperatures increase as is both witnessed in the model and experiments. Furthermore, varying rotational speed has most influence on HPA outlet temperature at low loop flow ratios, where EGR rate is generally lower. At higher flow ratios, EGR rate is high and the HPA temperature less sensitive to a variation in rotational speed. This effect is recorded in the simulation results through a reduced gradient.

Conclusions and Outlook

This paper presented a numerical model that extends previous one-dimensional codes to allow computation of torque generation in curved rotor passages through additional source terms that compute inviscid blade forces. The model was furthermore validated through both literature data and experimental data from a symmetrically cambered micro-wave rotor developed at the University of Bath. The main points of the paper can be summarised as follows:

- The objective of the study was to perform model validation through literature data for straight channeled pressure exchangers run at low temperature and through experimental data obtained from a symmetrically cambered micro-wave rotor developed at the University of Bath.
- The comparison with literature data for pressure exchangers at constant speed and inlet conditions yields good agreement for both steady and unsteady flow data. This gives confidence that the model is able to predict fundamental characteristics, such as wave patterns and port flow conditions. It further emphasises the importance of loss mechanisms on the performance characteristics of wave rotors.

- A comparison of the model with sub-1kW micro-wave rotor turbine data yielded promising results and showed that the model is able to identify the variation of pressure ratio and temperature with rotational speed. Predicted power output was also in very good agreement with the experimental data.

Nonetheless, a few restrictions in the model persist and can be listed as follows:

- The model appears to react more sensitively to changes in inlet temperature and leakage gap than witnessed in the experiments.
- Increases in nominal leakage lead to a larger relative error between the determined total pressure ratio for experiments and simulation. The issue is exacerbated if only the nominal, rather than the actual leakage gap for a thermally stable operating point are known rendering the determination of leakage coefficients difficult and results more deviating from experimentally determined ones.
- Power predictions are slightly overpredicted throughout, but show similar trends as in the experiments and vary only marginally with respect to changes in rotational speed. However, an additional loss factor is required to account for the (mis-)alignment of the flow at the port/rotor interface. For higher loop flow ratios, the increase in power output in the simulation results is lower compared to the experiments, resulting in a slight underprediction for $\lambda=2.2$.
- The model is able to predict port outlet temperatures well, but fails to identify the effects larger leakage gaps incur. The effect of reduced EGR rates and thus more fresh air being directed through the HPA port appears to be more pronounced than witnessed in the experiments. Also, ingestion of air from neighbouring channels and the leakage cavity seem to become more pronounced at elevated leakage gaps, which cannot be identified by the model. However, trends showing effect of increased loop flow ratio on the outlet temperature are captured well.
- The introduced model facilitates wave rotor turbine design process, as it provides the means to investigate both the pressure exchange capabilities of the wave rotor as well as its use as a power turbine. It can thus be used for initial sizing and as an inexpensive design tool to vary channel camber before more expensive CFD modeling tools are applied.

Acknowledgements

The authors would like to thank the Defence Science and Technology Laboratory (DSTL) for funding the project under the Defence and Security Accelerator *Beyond Battery Power* competition.

References

- [1] J. Wilson and D.E. Paxson, 1993. "Jet Engine Performance Enhancement Through Use of a Wave-Rotor Topping Cycle". *NASA Technical Memorandum 4486*.
- [2] E. Zauner, Y.-P. Chyou, F. W., and Althaus, R., 1993. "Gas Turbine Topping Stage Based on Energy Exchangers: Process and Performance". In ASME 1993 International Gas Turbine and Aeroengine Congress and Exposition, Volume 2: Combustion and Fuels; Oil and Gas Applications; Cycle Innovations; Heat Transfer; Electric Power; Industrial and Cogeneration; Ceramics; Structures and Dynamics; Controls, Diagnostics and Instrumentation, ASME.
- [3] P.H. Snyder and R.E. Fish, 1996. "Assessment of a Wave Rotor Topped Demonstrator Gas Turbine Engine Concepts". In ASME 1996 International Gas Turbine and Aeroengine Congress and Exhibition, Volume 1: Turbomachinery, ASME.
- [4] G.E. Welch, 1997. "Wave Engine Topping Cycle Assessment". In 35th Aerospace Sciences Meeting & Exhibit, AIAA.
- [5] G.E. Welch and S.M. Jones and D.E. Paxson, 1997. "Wave-Rotor-Enhanced Gas Turbine Engines". *Journal of Engineering for Gas Turbines and Power*, **119**, pp. 469–477.
- [6] T. Nagashima and K. Okamoto and Y. Ribaud, 2005. Cycles and Thermal System Integration Issues of Ultra-Micro Gas Turbines. Educational Notes RTO-EN-AVT-131, Neuilly-sur-Seine, France: RTO.
- [7] P. Akbari, R. Nalim, N. Müller, 2006. "Performance enhancement of microturbine engines topped with wave rotors". *Journal of Engineering for Gas Turbines and Power*, **128**(1), pp. 190–202.
- [8] G. Lenoble and S. Ogaji, 2010. "Performance Analysis and Optimization of a Gas Turbine Cycle Integrated with an Internal Combustion Wave Rotor". *Proceedings of the Institution of Mechanical Engineers, Part A: Journal of Power and Energy*, **224**(7), pp. 889–900.
- [9] L. Guzzella, U. Wenger and R. Martin, 2000. "IC-Engine Downsizing and Pressure-Wave Supercharging for Fuel Economy". *SAE Technical Paper*, 2000-01-1019.
- [10] Y. Oguri, T. Suzuki, M. Yoshida and M. Cho, 2001. "Research on Adaption of Pressure Wave Supercharger (PWS) to Gasoline Engine". *SAE Technical Paper*, 2004-01-0368.
- [11] F. Weber, L. Guzzella and C. Onder, 2002. "Modelling of a Pressure Wave Supercharger Including External Exhaust Gas Recirculation". *Proceedings of the Institution of Mechanical Engineers, Part D: Journal of Automobile Engineering*, **216**(3), pp. 217–235.
- [12] T. Suzuki, Y. Oguri, K. U., and Yoshida, M., 2004. "Experimental Investigation of Pressure Wave Supercharging for SI Engine". In ASME 2004 International Mechanical Engineering Congress and Exposition Process Industries, ASME.
- [13] P. Spring and C.H. Onder and L. Guzzella, 2007. "EGR Control of Pressure-Wave Supercharger IC Engines".

Control Engineering Practice, **15**, pp. 1520–1532.

- [14] L. Yan, Z. Dasen, Z. Hongguang, J. Changwei, L. Jiangguo and Z. Tong, 2008. “Experiment and CFD Investigation of Pressure-Wave Supercharger”. *SAE Technical Paper*, 2008-01-1631.
- [15] L. Pohořelský, J. Macek, M. Polášek and O. Vítek, 2004. “Simulation of a COMPREX Pressure Exchanger in a 1-D Code”. *SAE Technical Paper*, 2004-01-1000.
- [16] Y. Lei and D.S. Zhou and H.G. Zhang, 2010. “Investigation on Performance of a Compression-Ignition Engine with Pressure-Wave Supercharger”. *Energy*, **35**, pp. 85–93.
- [17] J. Zhao and D. Hu, 2017. “An Improved Wave Rotor Refrigerator Using an Outside Gas Flow for Recycling the Expansion Work”. *Shock Waves*, **27**(2), pp. 325–332.
- [18] D. Hu, R. Li, P. Liu and J. Zhao, 2016. “The Loss in Charge Process and Effects on Performance of Wave Rotor Refrigerator”. *International Journal of Heat and Mass Transfer*, **100**, pp. 497–507.
- [19] D. Hu, R. Li, P. Liu and J. Zhao, 2016. “The Design and Influence of Port Arrangement on an Improved Wave Rotor Refrigerator Performance”. *Applied Thermal Engineering*, **107**, pp. 207–217.
- [20] K. Pekkan and M.R. Nalim, 2003. “Two-Dimensional Flow and NO_x Emissions in Deflagrative Internal Combustion Wave Rotor Configurations”. *J. Eng. Gas Turbines Power*, **125**(3), pp. 720–730.
- [21] M.R. Nalim, Z.A. Izzy and P. Akbari, 2012. “Rotary Wave-Ejector Enhanced Pulse Detonation Engine”. *Shock Waves*, **22**(1), pp. 23–38.
- [22] M. Rajagopal, A. K., and R. Nalim, 2012. “Wave-Rotor Pressure-Gain Combustion Analysis for Power Generation and Gas Turbine Applications”. In ASME 2012 Gas Turbine India Conference, ASME.
- [23] R.D. Pearson, 1985. “A Gas Wave-Turbine Engine Which Developed 35 HP and Performed Over a 6:1 Speed Ranges”. In Proceedings ONR/NAVAIR Wave Rotor Research and Technology Workshop, Naval Postgraduate School.
- [24] R.D. Pearson, 1986. “Thermodynamics and Gas Dynamics of Internal Combustion Engines, Volume II”. In *The Oxford Handbook of Innovation*, D.E. Winterbone and S.C. Low, ed. Oxford University Press, Oxford.
- [25] A. Mathur, 1985. “Design and Experimental Verification of Wave Rotor Cycles”. In Proceedings ONR/NAVAIR Wave Rotor Research and Technology Workshop, Naval Postgraduate School.
- [26] R. Taussig, P. Cassady, J. Z. W. T., and Klostermann, E., 1983. Investigation of Wave Rotor Turbofans for Cruise Missile Engines. Tech. rep., Final Report Submitted by MSNW to DARPA.
- [27] H.E. Weber, 1996. *Shock Wave Engine Design*. John Wiley & Sons, New York.
- [28] A. Mathur and R.P. Shreeve, 1987. “Calculation of Unsteady Flow Processes in Wave Rotors”. *AIAA Paper* 87-0011.
- [29] A. Mathur, 1986. Code Development for Turbofan Engine Cycle Performance With and Without a Wave Rotor Component. Tech. rep., Report NPS67-86-006CR, Naval Postgraduate School, Monterey, CA.
- [30] J.W. Roberts, 1990. “Further Calculations of the Performance of Turbofan Engines Incorporating a Wave Rotor”. MS Thesis, Naval Postgraduate School, Monterey.
- [31] T.F. Salacka, 1985. “Review, Implementation and Test of the QAZID Computational Method with a View to Wave Rotor Applications”. MS Thesis, Naval Postgraduate School, Monterey.
- [32] D.T. Johnston, 1987. “Further Development of a One-Dimensional Unsteady Euler Code for Wave Rotor Applications”. MS Thesis, Naval Postgraduate School, Monterey.
- [33] K. Okamoto and T. Nagashima, 2003. A Simple Numerical Approach of Micro Wave Rotor Gasdynamic Design. Tech. rep., 16th International Symposium on Airbreathing Engines, Paper ISABE-2003-1213.
- [34] K. Okamoto, T. Nagashima and K. Yamaguchi, 2003. “Introductory Investigation of Micro Wave Rotor”. In International Gas Turbine Congress 2003 Tokyo.
- [35] A. Fatsis, A. Lafond and Y. Ribaud, 1998. “Preliminary Analysis of the Flow inside a Three-Port Wave Rotor by means of a Numerical Models”. *Aerospace Science and Technology*, **5**, pp. 289–300.
- [36] A. Fatsis, N.G. Orfanoudakis, D.G. Pavlou, A. Panoutsopoulou and N. Vlachakis, 2006. “Unsteady Flow Modelling of a Pressure Wave Supercharger”. *Proceedings of the Institution of Mechanical Engineers Part D Journal of Automobile Engineering*, **220**(2), pp. 209–218.
- [37] F. Iancu, J. Piechna and N. Müller, 2005. “Numerical Solutions for Ultra-Micro Wave Rotors (U μ WR)”. In Proceedings of the 35th International Symposium AIAA Fluid Dynamics Conference and Exhibit, AIAA.
- [38] D.E. Paxson, 1992. “A General Numerical Model for Wave Rotor Analysis”. *NASA-TM-105740*.
- [39] Paxson, D., and Wilson, J., 1993. “An Improved Numerical Model for Wave Rotor Design and Analysis”. AIAA, Jan.
- [40] D.E. Paxson and J. Wilson, 1995. “Recent Improvements to and Validation of the One Dimensional NASA Wave Rotor Model”. *NASA-TM-106913*.
- [41] D.E. Paxson, 1995. “A Numerical Model for Dynamic Wave Rotor Analysis”. *AIAA-95-2800*.
- [42] G.E. Welch and D.E. Paxson, 1998. “Wave Turbine Analysis Tool Development”. *NASA/TM-1998-208485*.
- [43] S. Chan and H. Liu, 2017. “Mass-based design and optimization of wave rotors for gas turbine engine enhancement”. *Shock Waves*, **27**(2), pp. 313–324.
- [44] S. Chan, H. Liu, F. Xing and H. Song, 2018. “Wave Rotor Design Method With Three Steps Including Experimental Validation”. *J. Eng. Gas Turbines Power*, **140**(11), pp. 111201–1–111201–13.
- [45] A. Mustafa, R.F. Martinez-Botas, A. Pesiridis, M.S.

- Chiong and S. Rajoo, 2014. "Assessment of Turbocharger Turbine Unsteady Flow Modelling Methodology on Engine Performance". *ASME*, July.
- [46] A. Costall, 2007. "A One-Dimensional Study of Unsteady Wave Propagation in Turbocharger Turbine". PhD Thesis, Imperial College London.
- [47] J.A.C. Kentfield, 1969. "The Performance of Pressure-Exchanger Dividers and Equalizers". *Journal of Basic Engineerin*, pp. 361–369.
- [48] J. Wilson and D. Fronek, 1993. "Initial results from the NASA-Lewis Wave Rotor Experiment". *AIAA-93-2521*.
- [49] D.W. Witte and K. Tatum and S.B. Williams, 1996. "Computation of Thermally Perfect Compressible Flow Properties". In 34th Aerospace Sciences Meeting & Exhibit, AIAA.
- [50] G.P. Smith, D.M. Golden, M. Frenklach, N.W. Moriarty, B. Eiteneer, M. Goldenberg, C.T. Bowman, R.K. Hanson, S. Song, W.C. Gardiner, Jr., V.V. Lissianski, and Z. Qin, The Gas Research Institute. GRI-Mech 3.0. URL http://www.me.berkeley.edu/gri_mech/.
- [51] C.R. Martin, 2010. HOT: Thermodynamic Tools for Matlab. Virginia Active Combustion Control Group. URL <http://hot-tdb.sourceforge.net/>.
- [52] D.E. Winterbone and R.J. Pearson, 2000. *Theory of Engine Manifold Design: Wave Action Methods for IC Engines*. Professional Engineering Publishing Limited.
- [53] J.A.C. Kentfield, 1993. *Nonsteady, One-Dimensional, Internal, Compressible Flows*. Oxford Engineering Sciences Series 31.
- [54] T.M. Kuzay and C.J. Scott, 1977. "Turbulent Heat Transfer Studies in Annulus with Inner Cyclinder Rotation". *Journal of Heat Transfer*, **99**(1), pp. 12–19.
- [55] L.M. Larosiliere, 1993. "Three-Dimensional Numerical Simulation of Gradual Opening in a Wave Rotor Passage". *AIAA-93-2526*.
- [56] Akbari, P., and Müller, N., 2003. "Preliminary Design Procedure for Gas Turbine Topping Reverse-Flow Wave Rotors". In International Gas Turbine Congress 2003 Tokyo.
- [57] Akbari, P., and Müller, N., 2003. "Gas Dynamic Design Analyses of Charging Zone for Reverse-Flow Pressure Wave Superchargers". In Proceedings of ICES03 2003 Spring Technical Conference of the ASME Internal Combustion Engine Division.
- [58] F. Iancu, J. Piechna and N. Müller, 2008. "Basic Design Scheme for Wave Rotors". *Shock Waves*, **18**(1), pp. 365–378.
- [59] J.D. Denton, 1993. "Loss Mechanisms in Turbomachines". *Journal of Turbomachinery*, **115**, pp. 621–656.